Model created in COMSOL Multiphysics 6.4

Hyperelastic Seal

Introduction

In this example you study the force–deflection relation of a car door seal made from a soft rubber material. The model uses a hyperelastic material together with formulations that can account for the large deformations and contact conditions. It is of special interest to investigate the effect of air confined within the seal.

See the *Structural Mechanics Module User's Guide* for theory about hyperelastic material.

Model Definition

The seal is compressed between a stationary plane surface and an indenting cylinder. There is also a vertical rigid wall at a distance of 1 mm from the initial position of the seal. Figure 1 shows the undeformed geometry of the seal and the contacting surfaces.

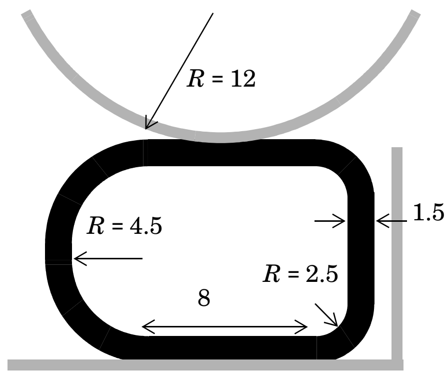


Figure 1: Model geometry.

The seal is modeled in 2D assuming plane strain conditions. The (arbitrary) thickness in the out-of-plane direction is 50 mm. The contacting surfaces are rigid when compared to the seal.

MATERIAL PROPERTIES

- The rubber is hyperelastic and modeled as a Mooney-Rivlin material with $C_{10} = 0.37$ MPa and $C_{01} = 0.11$ MPa. The material is almost incompressible, so the bulk modulus is set to 10^4 MPa. A mixed formulation is automatically used for this material model.
- The compression of the confined air is assumed to be adiabatic, giving the pressure-density relation

$$\frac{p}{p_{\mathrm{ref}}} = \left(\frac{\rho}{\rho_{\mathrm{ref}}}\right)^{\gamma} = \left(\frac{A_{\mathrm{ref}}}{A}\right)^{\gamma}$$

Here, the undeformed and deformed cross-section areas are denoted by A_{ref} and A, respectively. The ratio of specific heat, γ, has the value 1.4 and $p_{\mathrm{ref}} = 1$ atm is the reference air pressure. The load acting on the interior of the seal is then

$$\Delta p = p - p_{\mathrm{ref}} = p_{\mathrm{ref}}\left[\left(\frac{A_{\mathrm{ref}}}{A}\right)^{\gamma} - 1\right]$$

CONSTRAINTS AND LOADS

- One contact pair is used between the cylinder and the exterior seal boundaries.
- One contact pair is used between the stationary plates and the exterior seal boundaries.
- The lower straight part of the seal is glued to the car body. This is modeled with an adhesion condition.
- The rigid cylinder is lowered using a parameter in the parametric continuation solver that controls the negative y displacement. It starts with a gap of 0 mm and is lowered 4 mm.

Results and Discussion

Figure 2 shows the deformed shape at the lowest cylinder position — corresponding to an indentation of 4 mm — without internal pressure. The deformation scale is 1:1, that is, a true shape. The plot shows a detachment region of significant size.

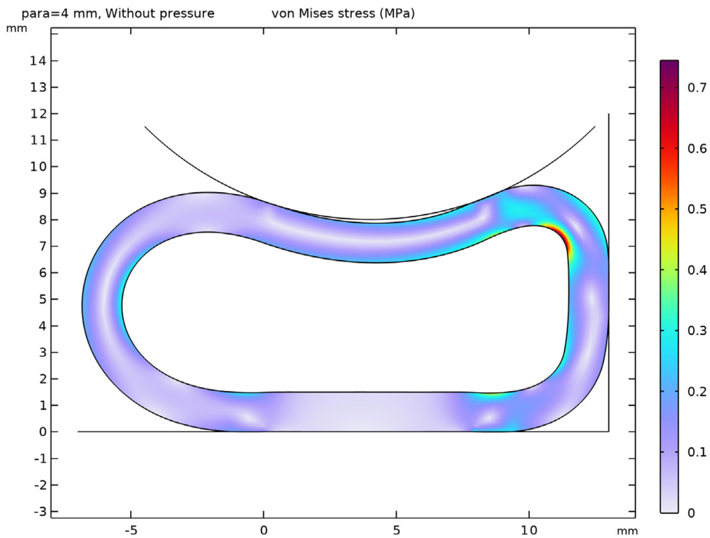


Figure 2: Seal deformation at 4 mm indentation without internal air pressure.

Figure 3 shows the corresponding contact pressure plot. The detachment region appears first at an indentation just over 2.5 mm and grows as the indentation increases further. The actual contact areas are reduced to two spots at the sides.

Such a significant change in the contact pressure distribution indicates that the computations must be performed using a fine mesh together with sufficiently small steps in the parametric analysis with respect to the indentation value.

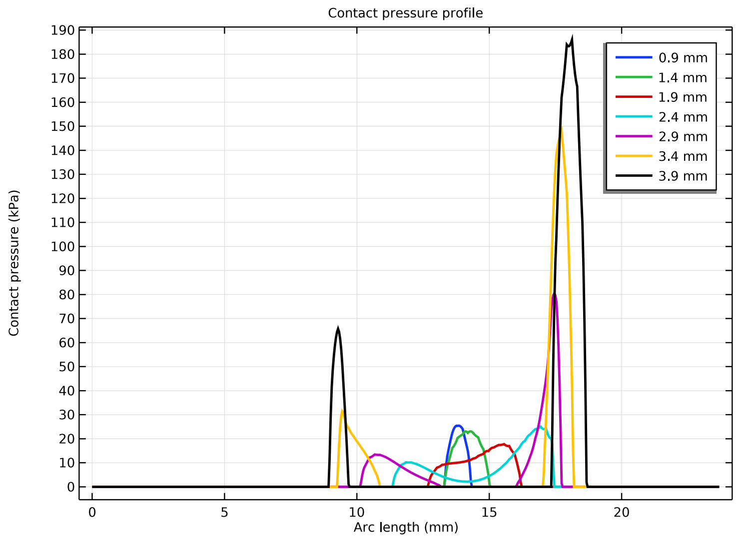


Figure 3: The contact pressure distribution over the area between the seal and cylinder for different indentations without internal air pressure.

Figure 4 shows the result of the computations with the internal pressure taken into account. The seal profile appears inflated. The contact pressure plot in Figure 5 confirms that there is no detachment region even though the contact pressure has a pronounced minimum in the middle part.

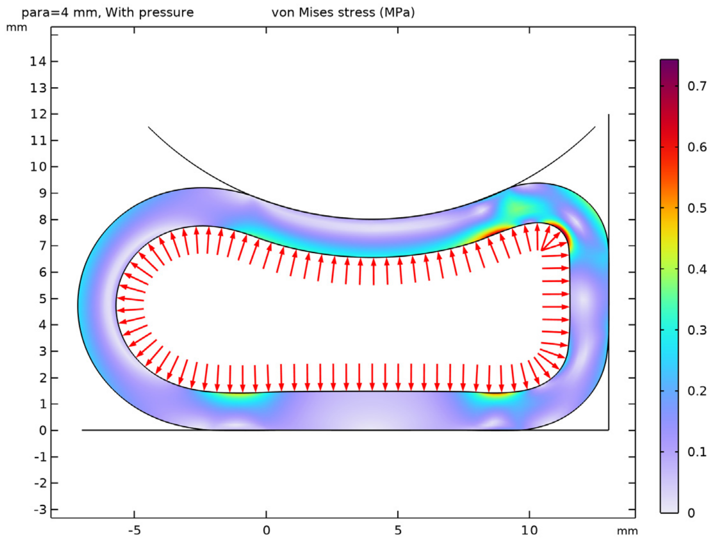


Figure 4: Seal deformation at 4 mm indentation with internal air pressure.

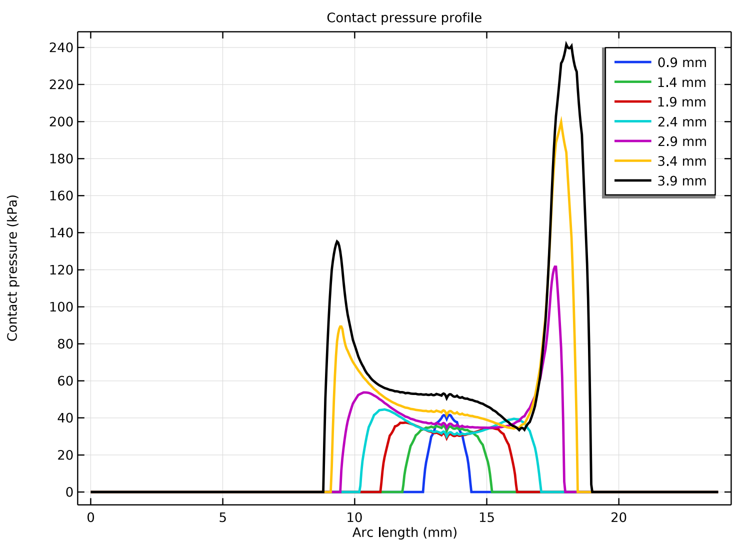


Figure 5: The contact pressure distribution for different indentations with internal air pressure taken into account.

Figure 6 shows a plot of the force per unit length versus the indentation of the rigid cylinder, with and without the internal pressure taken into account. The distinct change in slope of the curves is attributed to the rightmost part of the seal coming into contact with the vertical wall, so that the seal can no longer deform in that direction.

Notice that the forces needed to compress the seal can be almost one order of magnitude larger when the effect of the confined air is taken into account.

In reality, a car door seal contains small holes through which the air can escape as long as the compression is not too fast. Thus, the values computed with and without internal air pressure are the limits corresponding to very fast and very slow compression, respectively.

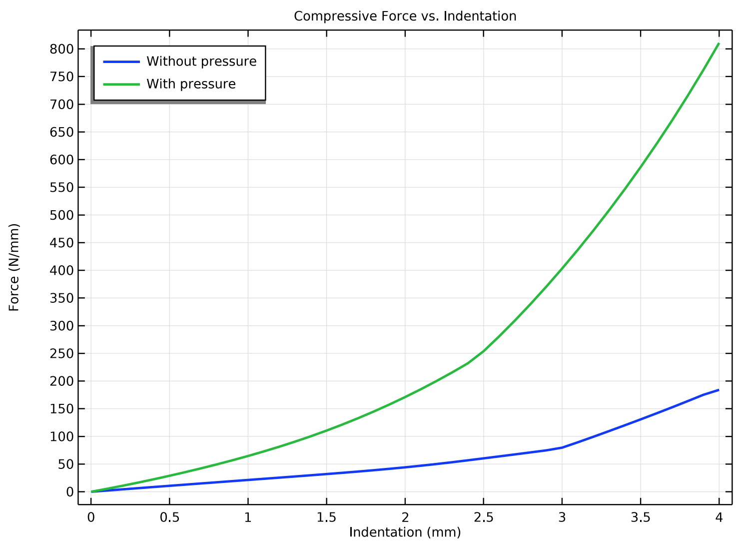


Figure 6: Compressive force per unit length versus indentation with and without internal pressure.

Notes About the COMSOL Implementation

The confined air inside the seal is modeled using an **Enclosed Cavity** node, which automatically computes the undeformed and deformed volume (or cross-section area) enclosed by the inner seal boundaries with the divergence theorem. The **Fluid** subnode adds the pressure load resulting from the volume change, in this case assuming the adiabatic compression of air.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/hyperelastic_seal`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

GEOMETRY 1

If you do not want to build all the geometry, you can load the geometry sequence from the stored model. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file `hyperelastic_seal.mph`. You can then continue to the **Global Definitions** section below.

To build the geometry from scratch, continue from here.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Width** text field, type 18.

5 In the **Height** text field, type 12.

6 Locate the **Position** section. In the **x** text field, type -6.

7 Click **Build Selected**.

Fillet 1 (fil1)

1 In the **Geometry** toolbar, click **Fillet**.

2 On the object **r1**, select Points 1 and 4 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type 6.

5 Click **Build Selected**.

Fillet 2 (fil2)

1 In the **Geometry** toolbar, click **Fillet**.

2 On the object **fil1**, select Points 4 and 5 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type 4.

5 Click **Build Selected**.

Thicken 1 (thi1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Thicken**.

2 Select the object **fil2** only.

3 In the **Settings** window for **Thicken**, locate the **Options** section.

4 From the **Offset** list, choose **Asymmetric**.

5 In the **Upside thickness** text field, type 1.5.

6 Click **Build Selected**.

Create the indenter.

Indenter

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, type Indenter in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type 12.

4 In the **Sector angle** text field, type 90.

5 Locate the **Position** section. In the **x** text field, type 4.

6 In the **y** text field, type 24.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-135`.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

9 From the **Show in physics** list, choose **Boundary selection**.

Create the support.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `20`.

4 Locate the **Position** section. In the **x** text field, type `-7`.

5 In the **y** text field, type `-1`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

7 From the **Show in physics** list, choose **Boundary selection**.

8 Find the **Cumulative selection** subsection. Click **New**.

9 In the **New Cumulative Selection** dialog, type `Rigid base` in the **Name** text field.

10 Click **OK**.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Height** text field, type `12`.

4 Locate the **Position** section. In the **x** text field, type `13`.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Rigid base**.

Convert to Curve 1 (ccur1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Curve**.

2 Select the objects **c1**, **r2**, and **r3** only.

3 In the **Settings** window for **Convert to Curve**, click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **ccur1**, select Boundaries 1, 2, 4–6, and 8–10 only.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

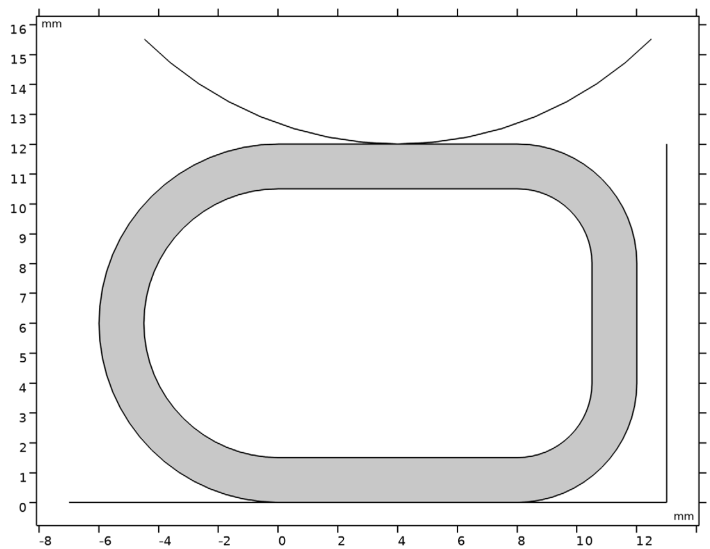
3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** checkbox.

5 Click **Build Selected**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

The model geometry is now complete.



Inner Seal Boundary

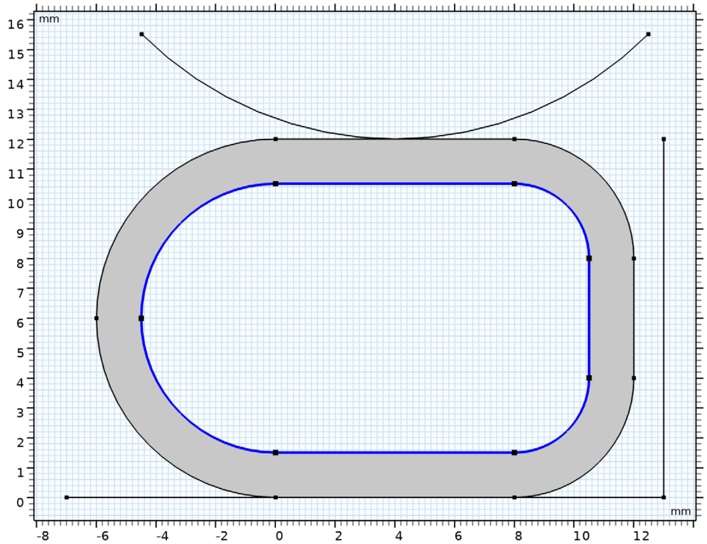
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Inner Seal Boundary` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** checkbox.

5 Select one of the boundaries on the inside of the seal.



Outer Seal Boundary

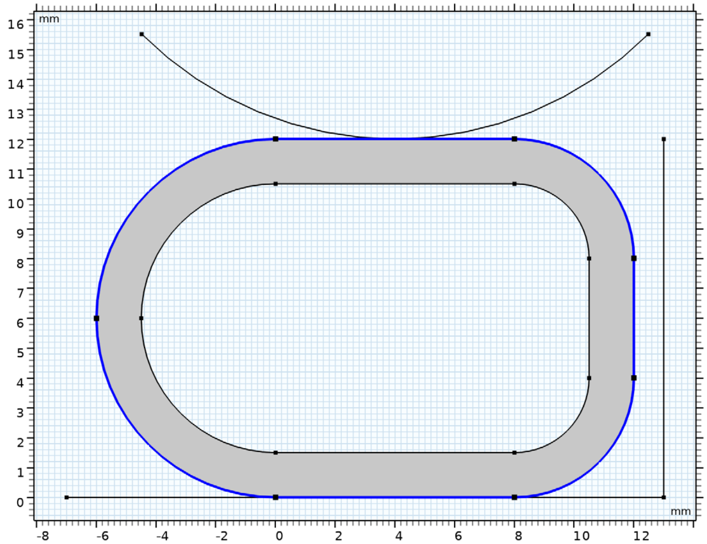
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Outer Seal Boundary` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** checkbox.

5 Select one of the boundaries on the outside of the seal.



Glued Seal Boundary

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Glued Seal Boundary` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundary 4 only.

Fillet 1 (fil1), Fillet 2 (fil2), Rectangle 1 (r1), Thicken 1 (thi1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1**, Ctrl-click to select **Rectangle 1 (r1)**, **Fillet 1 (fil1)**, **Fillet 2 (fil2)**, and **Thicken 1 (thi1)**.

2 Right-click and choose **Group**.

Seal

In the **Settings** window for **Group**, type `Seal` in the **Label** text field.

GLOBAL DEFINITIONS

Add a parameter that you can use to gradually increase the vertical displacement.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`para`	`0[mm]`	0 m	`Vertical displacement parameter`
`d`	`50[mm]`	0.05 m	`Out-of-plane thickness`

DEFINITIONS

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, type `upper` in the **Pair name** text field.

3 Locate the **Source Boundaries** section. From the **Selection** list, choose **Indenter**.

4 Locate the **Destination Boundaries** section. From the **Selection** list, choose **Outer Seal Boundary**.

Contact Pair 2 (p2)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, type `lower` in the **Pair name** text field.

3 Locate the **Source Boundaries** section. From the **Selection** list, choose **Rigid base**.

4 Locate the **Destination Boundaries** section. From the **Selection** list, choose **Outer Seal Boundary**.

The boundaries in the contact pairs are unnecessarily large because it was convenient to reuse existing selections. In large 3D models, you should however keep down the size of the contact boundaries for performance reasons.

Since the indenter is only modeled as a rigid boundary, prescribe its deformation using a moving mesh. Alternatively, the indenter could be modeled as a rigid or elastic domain in the Solid Mechanics interface, in which case its deformation would be prescribed in the interface.

COMPONENT 1 (COMP1)

Prescribed Deformation 1

1 In the **Physics** toolbar, click **Moving Mesh** and choose **Prescribed Deformation**.

2 In the **Settings** window for **Prescribed Deformation**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Indenter**.

5 Locate the **Prescribed Deformation** section. Specify the dx vector as

0	X
-para	Y

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.

3 In the d text field, type `d`. In the plane strain approximation, this setting only affects total force computations.

Hyperelastic Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.

2 In the **Settings** window for **Hyperelastic Material**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Mooney–Rivlin, two parameters**.

5 In the κ text field, type `1e4[MPa]`.

Contact 1

In the **Model Builder** window, click **Contact 1**.

Friction 1

1 In the **Physics** toolbar, click **Attributes** and choose **Friction**.

2 In the **Settings** window for **Friction**, locate the **Friction Parameters** section.

3 In the μ text field, type `0.3`.

Add an adhesion condition to model the glue layer at the bottom of the seal.

Contact 1

In the **Model Builder** window, click **Contact 1**.

Adhesion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Adhesion**.

2 In the **Settings** window for **Adhesion**, locate the **Adhesive Activation** section.

3 From the **Activation criterion** list, choose **User defined**.

4 In the text field, type `dom==4`.

5 Locate the **Adhesive Stiffness** section. From the **Adhesive stiffness** list, choose **User defined**.

6 Specify the **k** vector as

1e10[N/m^3]	t1
2e10[N/m^3]	n

Add an **Enclosed Cavity** node to model the effect of air being compressed inside the seal.

Enclosed Cavity 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Enclosed Cavity**.

2 In the **Settings** window for **Enclosed Cavity**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inner Seal Boundary**.

Fluid 1

Inspect the **Fluid** subnode, which defines the properties of the gas being compressed. To study the effect of the confined air, add the **Fluid** node to a load group.

1 In the **Model Builder** window, click **Fluid 1**.

2 In the **Physics** toolbar, click **Load Group** and choose **New Load Group**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Model parameters	C10	0.37[MPa]	Pa	Mooney-Rivlin
Model parameters	C01	0.11[MPa]	Pa	Mooney-Rivlin
Density	rho	1100[kg/m^3]	kg/m³	Basic

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

2 Select Boundaries 1–3 only.

Distribution 1

1 Right-click **Edge 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type `1`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Rigid base**.

Distribution 2

1 In the **Model Builder** window, right-click **Edge 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type `50`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Indenter**.

Free Quad 1

In the **Mesh** toolbar, click **Free Quad**.

Size 1

1 Right-click **Free Quad 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Inner Seal Boundary**.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `0.2`.

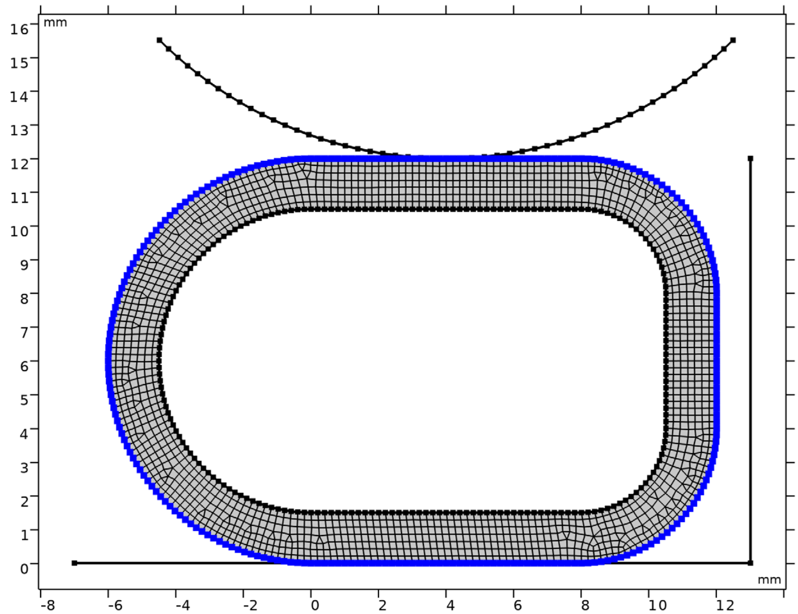
Size 2

1 Right-click **Size 1** and choose **Duplicate**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Outer Seal Boundary**.

4 Click **Build All**.



STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the parameter `para`. Start at a nonzero value to avoid ill-conditioning during contact initiation.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Vertical displacement parameter)	`1e-3 range(0.1,0.1,4)`	`mm`

Define two load cases to study the effect of the confined air.

6 Select the **Define load cases** checkbox.

7 Click **Add**.

8 Click Add.

9 In the table, enter the following settings:

Load case	lgI	Weight
Without pressure		1.0
With pressure	√	1.0

10 In the Study toolbar, click Get Initial Value.

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

8 Click **Add Physical Quantity**.

9 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Face load (N/m^2)** in the tree.

10 Click **OK**.

11 In the **Settings** window for **Preferred Units**, locate the **Units** section.

12 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Face load	N/m^2	kPa

13 Click **Apply**.

Stress (solid)

Add an **Arrow Line** plot to visualize the gas pressure.

Arrow Line 1

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Enclosed cavities** > **Enclosed Cavity 1** > **solid.enc1.fax,solid.enc1.fay - Force per deformed area (spatial frame)**.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `300`.

5 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

7 Clear the **Color** checkbox.

8 Clear the **Color and data range** checkbox.

STUDY 1

Solver Configurations

In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** node.

The default scale for the displacement variables is calculated from the entire geometry size. For models with prescribed displacements as domain or boundary constraints, the maximum prescribed displacement usually gives a better estimate of the scale.

2 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** node, then click **Displacement Field (comp1.u)**.

3 In the **Settings** window for **Field**, locate the **Scaling** section.

4 In the **Scale** text field, type `1e-3`.

Change the scale for the auxiliary pressure to account for the material properties of the seal made of soft rubber.

5 In the **Model Builder** window, click **Auxiliary Pressure (comp1.solid.hmm1.pw)**.

6 In the **Settings** window for **Field**, locate the **Scaling** section.

7 In the **Scale** text field, type `1e5`.

8 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node, then click **Direct**.

9 In the **Settings** window for **Direct**, locate the **General** section.

10 From the **Solver** list, choose **PARDISO**.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Results While Solving** section.

3 Select the **Plot** checkbox.

4 In the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

2 In the **Settings** window for **2D Plot Group**, click next to **cycle_plot_level**, then choose **Load case**.

3 Click **Plot First**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

The default plot shows the von Mises stress distribution in the seal. The case without confined air is shown in Figure 2.

Next, visualize the stress distribution with confined air.

5 Click **Plot Last**.

You can see that the detachment region has disappeared as a result of the seal pressurization, compare with Figure 4.

Add a plot from **Result Templates** to visualize the normal and tangential contact forces on the seal.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Contact Forces (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Contact Forces (solid)

In the **Model Builder** window, expand the **Results** > **Contact Forces (solid)** node.

Color Expression

1 In the **Model Builder** window, expand the **Results** > **Contact Forces (solid)** > **Contact 1, Pressure** node, then click **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Expression** section.

3 From the **Unit** list, choose **kPa**.

Color Expression

1 In the **Model Builder** window, expand the **Results** > **Contact Forces (solid)** > **Contact 1, Friction Force** node, then click **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Expression** section.

3 From the **Unit** list, choose **kPa**.

The following steps show how to display the contact pressure at the bottom of the seal.

Contact Pressure

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Contact Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (Load case)** list, choose **First**.

4 From the **Parameter selection (para)** list, choose **Manual**.

5 In the **Parameter indices (1-41)** text field, type `range(10,5,40)`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Contact pressure profile`.

Line Graph 1

1 Right-click **Contact Pressure** and choose **Line Graph**.

2 Select Boundaries 7, 11, and 17 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Contact** > **solid.Tn - Contact pressure - N/m²**.

4 Click to expand the **Legends** section. Select the **Show legends** checkbox.

5 Find the **Include** subsection. Clear the **Solution** checkbox.

6 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `eval(para) mm`.

7 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

8 In the **Contact Pressure** toolbar, click **Plot**.

The plot in the **Graphics** window should now look like that in Figure 3.

Contact Pressure

Next, plot the pressure profile when the internal air pressure is included.

1 In the **Model Builder** window, click **Contact Pressure**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (Load case)** list, choose **Last**.

4 In the **Contact Pressure** toolbar, click **Plot**.

Finally, compute the force needed for the compression as the sum of all vertical reaction forces on the indenter.

Compressive Force vs. Indentation

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Compressive Force vs. Indentation` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

Global 1

1 In the **Compressive Force vs. Indentation** toolbar, click **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`-solid.dcnt1.T_toty_upper/d`	`N/m`	

4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

Compressive Force vs. Indentation

1 In the **Model Builder** window, click **Compressive Force vs. Indentation**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** checkbox.

4 Select the **y-axis label** checkbox.

5 In the **x-axis label** text field, type `Indentation (mm)`.

6 In the **y-axis label** text field, type `Force (N/mm)`.

7 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Compare with the plot shown in Figure 6.